Setting Up and Running FEAMAC

The following two files are required to run FEAMAC:

1) `feamac.lib` – Static FORTRAN library of FEAMAC code
2) `feamac.for` – FORTRAN 77 source code compiled at run-time by ABAQUS

`feamac.for` contains the ABAQUS `UMAT` and `UEXPAN` subroutines that call the FEAMAC code. This code is provided in compiled form as the static FORTRAN library `feamac.lib`. The following steps should be taken to allow ABAQUS to access these two files.

- Include path to `feamac.lib` in ABAQUS environment file (see below)
- Place `feamac.for` in ABAQUS working directory
- Run ABAQUS using the command:

```
abaqus –j job –user feamac interactive
```

This will cause ABAQUS to compile the `feamac.for` code and link it with the ABAQUS analysis code and the `feamac.lib` code. In the above, *`job`* represents the ABAQUS job, or input file, name (with no extension). This file is a standard ABAQUS input file, but with a few required specifications to indicate that an FEAMAC job is to be executed.

An example ABAQUS environment file in which the path to the FEAMAC library `feamac.lib` has been specified (highlighted) is given below. Note that the correct path to the library file on your computer should be specified.

Abaqus_v6.env

```
#
#        System-Wide ABAQUS Environment File
#       -------------------------------------
pre_memory = "256 mb"
standard_memory = "800 mb"
#
# NT specific settings
#
# Compile and link for user subroutines
# Compile_cpp and link_exe for ABAQUS/Make.
#
compile_fortran=['df', '/c', '/nologo', '/include:%I']
compile_cpp=('cl', '/c', '/nologo', '/W0', '/MD', '/TP', '/DNDEBUG', '/DWIN32',
             '/DTP_IP', '/D_CONSOLE', '/DNTI', '/DFLT_LIC', '/DOL_DOC',
             '/D__LIB__', '/DHKS_NT', '', '/DFAR=', '/D_WINDOWS',
             '/O1', '/I%I')

link_sl='LINK /debug /nologo /subsystem:console /INCREMENTAL:NO /defaultlib:dformd.lib
/nodefaultlib:dfor.lib /defaultlib:msvcrt.lib /nodefaultlib:msvcrtd.lib /nodefaultlib:libcd.lib
/nodefaultlib:libc.lib /dll /def:%E /out:%U %F %A %B oldnames.lib user32.lib ws2_32.lib
netapi32.lib advapi32.lib C:/mac/feamac/feamac.lib'

link_exe='LINK /nologo /subsystem:console /INCREMENTAL:NO /defaultlib:dformd.lib
/nodefaultlib:dfor.lib /defaultlib:msvcrt.lib /nodefaultlib:msvcrtd.lib /nodefaultlib:libcd.lib
/nodefaultlib:libc.lib /out:%J %F %M %L %B %O oldnames.lib user32.lib ws2_32.lib netapi32.lib
advapi32.lib'

import driverUtils, os
graphicsEnv = driverUtils.locateFile(os.environ['ABA_PATH'],'site','graphicsConfig','env')
if graphicsEnv:
    execfile(graphicsEnv)
else:
    raise 'Cannot find the graphics configuration environment file (graphicsConfig.env)'

del driverUtils, os, graphicsEnv
```

```
abaquslm_license_file="@aix.license.virginia.edu"
max_history_requests=0

doc_root="http://Bednarcyk:2080/v6.4"
```

ABAQUS Input File Requirements

The ABAQUS input file requires three additional specifications for FEAMAC jobs. First, a material specification must be made that indicates a MAC/GMC 4.0 input file to describe the material behavior, second, an orientation for the heterogeneous material must be specified, and third, a variable initialization specification must be made.

MAC/GMC Material Specification

The following is an example of a MAC/GMC material specification within an ABAQUS input file:

```
*MATERIAL,NAME=DOGL.MAC
*USER MATERIAL, CONSTANTS=0
*DEPVAR
207
*USER DEFINED FIELD
*EXPANSION, TYPE=ANISO, USER
```

The material name specified in this case is `DOGL.MAC`. The "`.MAC`" extension indicates to FEAMAC that this material is a MAC/GMC material and that the MAC/GMC heterogeneous material specification data is located in the file `DOGL.MAC`. This file must be placed in the ABAQUS working directory. It must be in the correct MAC/GMC 4.0 format and will be checked for compliance by the FEAMAC code.

The `*USER MATERIAL` specification is used for inputting material constants for user-defined constitutive models. In the case of FEAMAC, the material properties are obtained from the MAC/GMC input file. Thus, while the `*USER MATERIAL` specification is required by ABAQUS, it is not used by FEAMAC, and the number of constants should always be set to 0 (`CONSTANTS=0`).

The `*DEPVAR` specification is used to allocate space at each integration point for solution-dependent state variables. In the case of FEAMAC, this number of solution-dependent state variables is directly related to the number of subcells in the MAC/GMC material. In the case of standard (non-electromagnetic) analysis, this number of solution-dependent state variables is given by,

$$\text{DEPVAR} = 39 + 42\, N_\alpha\, N_\beta\, N_\gamma \left(N_{\text{int}}\right)^2$$

where N_α, N_β, and N_γ are the number of subcells in the x_1-, x_2-, and x_3-directions, respectively, within the specified repeating unit cell, and N_{int} is the number of integration points in an HFGMC analysis ($N_{\text{int}} = 1$ for GMC analyses). In the case of electromagnetic GMC analyses,

$$\text{DEPVAR} = 57 + 60\, N_\alpha\, N_\beta\, N_\gamma$$

Note that electromagnetic analysis is not currently available with HFGMC.

The `*USER DEFINED FIELD` specification indicates that, for integration points containing this material, the ABAQUS user-definable subroutine `USDFLD` will be called. This subroutine can call the ABAQUS `GETVRM` subroutine to obtain field variables throughout the execution. The `USDFLD` subroutine is provided to FEAMAC users in the `feamac.for` file. If use of the `USDFLD` subroutine is not desired, the `*USER DEFINED FIELD` specification can be omitted.

The `*EXPANSION` specification is used to define the thermal expansion coefficients for the material. The `USER` specification indicates that the thermal expansion behavior will be determined via MAC/GMC through the ABAQUS `UEXPAN` user-definable subroutine. `TYPE=ANISO` indicates that the thermal expansion of the MAC/GMC material can, in general, be anisotropic. The `*EXPANSION` specification should be used exactly as shown in the above example when the FEAMAC problem involves a temperature change. In the isothermal case, the `*EXPANSION` specification can be omitted.

Orientation Specification

Specifying a material orientation in FEAMAC is done in exactly the same way as in all ABAQUS problems that employ anisotropic materials. Examples of orientation specifications within an ABAQUS input file are,

```
*ORIENTATION, NAME=LONG
1.0, 0.0, 0.0, 0.0, 1.0, 0.0
*ORIENTATION, NAME=TRANS
0.0, 1.0, 0.0, -1.0, 0.0, 0.0
```

Here, the orientation named "`LONG`" aligns the local material coordinate directions with the global FEA coordinate directions. The first three numbers define a point on the x-axis of the local coordinate system with respect to the global coordinate system, while the second three numbers define a point on the y-axis of the local coordinate system with respect to the global coordinate system. The second example above specifies an orientation named "`TRANS`" whose local x-direction is aligned with the global y-direction, and whose local y-direction is aligned with the global negative x-direction. For more information see the ABAQUS documentation. As is the case in all ABAQUS problems, the material and orientation must be associated with an element set through the `*SOLID SECTION` specification.

Variable Initialization Specification

Two initialization specifications are typically used within FEAMAC:

```
*INITIAL CONDITIONS, TYPE=TEMPERATURE
ALL, 23.0
*INITIAL CONDITIONS, TYPE=SOLUTION, USER
```

The `*INITIAL CONDITIONS, TYPE=TEMPERATURE` specification indicates to ABAQUS the starting temperature for the problem. MAC/GMC can employ temperature-dependent material properties, it is thus important, even when executing a problem with no thermal loading, to specify a temperature for the problem. The first line after this specification (`ALL, 23.0`) indicates the node set (`ALL`) and the initial temperature for the node set (`23.0`). If the initial temperature is not specified, FEAMAC will assume a starting temperature of 0.0.

The `*INITIAL CONDITIONS, TYPE=SOLUTION, USER` specification indicates to ABAQUS that the solution-dependent state variable initialization subroutine `SDVINI` should be executed at the beginning of the FEAMAC problem. This user-definable subroutine, contained in `feamac.for`, performs initialization tasks, and this specification should always be used for FEAMAC problems exactly as shown in the above example.

MAC/GMC 4.0 Input File Requirements

The format of the MAC/GMC 4.0 input file for use within FEAMAC problems is identical to the form required for generation of effective material property data within standalone MAC/GMC 4.0 cases.

That is, the MAC/GMC 4.0 input file must specify the information about the heterogeneous material, but not information about any applied loading. This is because, for FEAMAC problems, the loading on the MAC/GMC repeating unit cell is determined from the local strain field at each integration point, which is passed to the MAC/GMC code from ABAQUS.

Table 1 provides a list of the 22 MAC/GMC 4.0 keywords and their association with FEAMAC. As indicated, the five keywords that involve applying loading to a repeating unit cell or laminate cannot be used within FEAMAC. If these keywords are present, they will be ignored. A minimum of three MAC/GMC 4.0 keywords are required to define completely a heterogeneous material for use in FEAMAC problems. These are: 1) `*CONSTITUENTS`; 2) One of either `*RUC` or `*LAMINATE`; and 3) `*END`. These three keywords allow the specification of the constituent materials of the heterogeneous material, the arrangement of the materials in a repeating unit cell or laminate configuration, and the end of the MAC/GMC 4.0 input file. Many additional MAC/GMC 4.0 keywords are optional within FEAMAC problems. However, in this beta release of FEAMAC, a good number of these optional keywords have not yet been tested. Thus, it is highly likely that an error will result if an FEAMAC problem is attempted with these optional MAC/GMC 4.0 keywords active. If you attempt such a problem, it is requested that you report the problem and any results to the contacts given at the beginning of this document, even if the results seem to be correct. This will aid in testing and debugging FEAMAC in pursuit of the final release version of the code.

One new MAC/GMC keyword, specifically for use in FEAMAC has been added: `*CONVERT`. This allows the specification of a stress and stiffness conversion factor for the results passed from MAC/GMC to ABAQUS. This enables the use of different stress and stiffness units in MAC/GMC and FEAMAC within the same FEAMAC problem. For example, if the MAC/GMC input file employs material properties that are in units of ksi, while the ABAQUS input file uses stress units of MPa and length units of mm, one can simply set the conversion factor to 6.895 MPa/ksi and the problem will execute correctly with ABAQUS-level results in MPa and MAC/GMC-level results in ksi. Each MAC/GMC input file can have its own conversion factor. The usage of `*CONVERT` is:

```
*CONVERT
  FACTOR=factor
```

where *factor* is the conversion factor by which the MAC/GMC stress and stiffness components are multiplied before being passed to ABAQUS. See Example Problem 4 for an example of the usage.

Table 1. MAC/GMC 4.0 keywords and their association with FEAMAC.

Keyword	Required for FEAMAC?	Optional in FEAMAC?	Tested in FEAMAC?
Flag-Type Keywords			
*CHECK	No	Yes	No
*CONDUCTIVITY	No	Yes	No
*ELECTROMAG	No	Yes	No
Material Keywords			
*MDBPATH	No	Yes	No
*CONSTITUENTS	Yes	Yes	Yes
Analysis Type and Architecture Keywords			
*RUC	No†	Yes	Yes
*LAMINATE	No†	Yes	No
Loading Keywords			
*MECH	No	No	–
*THERM	No	No	–
*SOLVER	No	No	–
*SURF	No	No	–

Keyword	Required for FEAMAC?	Optional in FEAMAC?	Tested in FEAMAC?
Damage and Failure Analysis Keywords			
*ALLOWABLES	No	Yes	No
*FAILURE_SUBCELL	No	Yes	No
*FAILURE_CELL	No	Yes	No
*DAMAGE	No	No	–
*DEBOND	No	Yes	Yes
*CURTIN	No	Yes	Yes
Results and Data Output Keywords			
*PRINT	No	Yes	Yes
*XYPLOT	No	Yes	Yes
*PATRAN	No	Yes	No
*MATLAB	No	Yes	No
Special FEAMAC Keyword			
*CONVERT	No	Yes	Yes
End of File Keyword			
*END	Yes	Yes	Yes

† Note that one of the two keyword *RUC or *LAMINATE is required.

An example MAC/GMC 4.0 input file for use in FEAMAC is given below. The first line is a title line for the file after which the keywords and associated input data are specified. Under *CONSTITUENTS five materials are specified, with materials 1, 2, and 5 having properties typed directly into the input file and materials 3 and 4 having properties taken from the MAC/GMC 4.0 materials database. *RUC defines the repeating unit cell architecture, while *DEBOND and *CURTIN define debonding and Curtin fiber breakage model data. *PRINT and *XYPLOT are associated with data output (see below), while *END indicates the end of the MAC/GMC 4.0 input file. Note that the special characters "#" and "&" represent a comment line and a line continuation, respectively.

```
DOG
*CONSTITUENTS
  NMATS=5
# -- Graphite fiber
  M=1 CMOD=6 MATID=U MATDB=1 &
   EL=388.2E3,7.6E3,0.41,0.45,14.9E3,-0.68E-6,9.74E-6
# -- Epoxy matrix
  M=2 CMOD=6 MATID=U MATDB=1 &
   EL=3.45E3,3.45E3,0.35,0.35,1.278E3,45.E-6,45.E-6
  M=3 CMOD=6 MATID=E
  M=4 CMOD=4 MATID=A
  M=5 CMOD=6 MATID=U MATDB=1 &
   EL=40.E3, 40E3, 0.32, 0.32, 15E3, 4.0E-6, 4.0E-6
*RUC
  MOD=2 ARCHID=11 VF=0.33 R=1. F=3 M=4
*DEBOND
  NII=1
  DBCH=1 NBI=1 NGI=1 FACE=3 BDN=35.0 LN=0.000004 BN=60. TOLN=0. &
  BDS=40 LS=0.1 BS=100. DELAY=0.
*CURTIN
  NCURT=1
  NBI=1 NGI=1 D=142.E-6 L0=0.0254 SIG0=609. TAU0=2.03 M=10.0 &
  DELAY=0. ACTION=1
*PRINT
  NPL=1
*XYPLOT
  FREQ=1
```

```
  MACRO=1
  NAME=dog X=2 Y=8
  MICRO=0
*END
```

Obtaining MAC/GMC 4.0 Output in FEAMAC

In the final release version of FEAMAC, all four MAC/GMC 4.0 results and data output keywords will be available for use in obtaining output (see Table 1). Currently, only the `*PRINT` and `*XYPLOT` have been tested. `*PRINT` allows specification of the overall print output level to the MAC/GMC 4.0 output file. In FEAMAC, each MAC/GMC 4.0 input file will generate an output file with the same name, but with the extension `.out`. It will contain echoed information from the MAC/GMC 4.0 input file, along with effective property results. If the MAC/GMC input file contains an error, it will be described in the `.out` file. Note that the third output file section, associated with time-based output in standalone MAC/GMC 4.0 problems, will not be written to the output file for FEAMAC problems. This is because each MAC/GMC 4.0 material specified in an FEAMAC problem can potentially be applied to many integration points within the ABAQUS FEA mesh. Thus, the time-dependent behavior is not unique to a particular MAC/GMC 4.0 material. Note also that, if `*PRINT` is omitted, a default print level of 1 will be employed.

The `*XYPLOT` keyword enables the generation of ASCII files of history-dependent data at the repeating unit cell and subcell level within FEAMAC. The use of the `*XYPLOT` keyword is identical to that used in standalone MAC/GMC 4.0 problems. As stated above, the history-dependent behavior within FEAMAC is not unique to particular MAC/GMC 4.0 material definition, rather, the behavior varies from element to element and integration point to integration point. For this reason, when an xy plot is specified for a material in an FEAMAC problem, an xy plot file will be generated for every element and integration point containing that material. In order to generate unique plot files, the element number and integration point number are appended to the plot file name. For example, the following xy plot specification within a MAC/GMC 4.0 input file,

```
*XYPLOT
  FREQ=1
  MACRO=1
  NAME=dog X=2 Y=8
  MICRO=0
```

would result in the generation of an xy plot file for every element and integration point containing the MAC/GMC 4.0 material specified in the input file. These plot files would be named, `DOG_macro_EL1_PT1.data`, `DOG_macro_EL1_PT2.data`, …, where the element and integration point number have been indicated. If xy plot files are desired for only a single element or group of elements, these elements should be assigned a unique MAC/GMC 4.0 material whose input file specifies the desired xy plot information. For the remaining elements, the `*XYPLOT` keyword can then be omitted, resulting in plot files being generated only for the desired elements.

In addition to utilizing MAC/GMC 4.0 keywords to generate local output at the ABAQUS integration points, the ABAQUS output capabilities can be employed. This is accomplished by specifying within the ABAQUS input file that solution-dependent state variable data is desired. For example, specifying,

```
*OUTPUT, FIELD, FREQUENCY=2
*ELEMENT OUTPUT, ELSET=EALL
SDV
```

within the ABAQUS input file indicates that the solution-dependent state variables (SDV) should be written to the ABAQUS output database for the elements in the element set EALL. The output database can then be accessed within ABAQUS/CAE (and other FEA post-processing tools) to generate contour plots and other data visualizations. The following indicates the numbering of the solution-dependent state variables (as they are accessed in ABAQUS/CAE by number),

```
        quantity      location in SDV
  -
  |     total strain      (1-6)
  |                                "macro" quantities when
  |     cauchy stress     (7-12)    micromechanics model
  |                                 or continuum model
  |     inelastic strain  (13-18)   is used
  |                                 NGVARS = 37 or 55
  |     state variable(s) (19-30)
        (space for 2 (6x1) vectors)
NGVARS
        thermal strain    (31-36)
  |
  |     temperature       (37)
  |
  |     electric field, E (38-40)
  |
  |     magnetic field, H (41-43)   NOTE: space for macro
  |                                 E-M quantities is present
  |     electric flux, D  (44-46)   only for ELECTROMAG problems
  |
  |     magnetic flux, B  (47-49)
  |
  |     thermal-electric  (50-52)
  |         field
  |
  |     thermal-magnetic  (53-55)
  |         field
  -
 --------------------------------
  -
  |     total strain      (56-61)  (1-6)
  |
  |     cauchy stress     (62-67)  (7-12)   "micro" quantities:
  |                                         this block of NSVARS = 42 or 60
  |     inelastic strain  (68-73)  (13-18)  is repeated for each subcell
  |                                         NSVARS is defined in SCANP.F90
  |     state variable(s) (74-85)  (19-30)
  |     (space for 2 [6x1] vectors)
  |
        thermal strain    (86-91)  (31-36)
NSVARS
        rn & rt debond    (92-97)  (37-42)
  |     parameters
  |
  |     electric field, E (98-100) (43-45)
  |
  |     magnetic field, H (101-103)(46-48)  NOTE: space for micro
  |                                         E-M quantities is present
  |     electric flux, D  (104-106)(49-51)  only for ELECTROMAG problems
  |
  |     magnetic flux, B  (107-109)(52-54)
  |
  |     thermal-electric  (110-112)(55-57)
  |         field
  |
  |     thermal-magnetic  (113-115)(58-60)
  |         field
 ---   .........................
  |              :
  .            etc.    (116-...)
  .
```

We see that the state variables for each integration point are stored in two sections. The first section, of length NGVARS, are global, or repeating unit cell level quantities. In the case of electromagnetic problems, the length of this section is NGVARS = 55, for standard thermo-mechanical (i.e., non-electromagnetic) problems, the length of this section is NGVARS = 37. The next section contains subcell quantities and is itself divided into subsections by subcell number. That is, after the global (repeating unit cell) level data comes data for subcell number 1, followed by data for subcell number 2, etc. The length of the section for each subcell is NSVARS, where, in the case of electromagnetic problems, the length of each section is NSVARS = 60, and for standard thermo-mechanical (i.e., non-electromagnetic) problems, the length of each section is NSVARS = 42. It is thus possible from the above information to locate any piece of information for the repeating unit cell or subcell stored within `SDV` in the ABAQUS/CAE output database.

In addition to the `SDV` output described above, three special quantities have been placed in locations 19, 20, and 21. As indicated above, locations 19-30 are reserved for unit cell level state variables, which are not currently used. These special quantities are given below.

Location in SDV	**Quantity**
19	Curtin fiber breakage model damage fraction for subcell 1,1
20	Repeating unit cell stiffness matrix component C_{11}
21	Repeating unit cell level equivalent plastic strain

It should be noted that writing the solution-dependent state variables to the ABAQUS output database will slow execution time as it typically will involve a large amount of data.